

# CFD test case 1

## Laminar oil transport in a pipeline

Suggestions for setting up the case

Ing. Luca Nicola Quaroni

Ing. Jorge Soto

Ing. Federico Lanteri

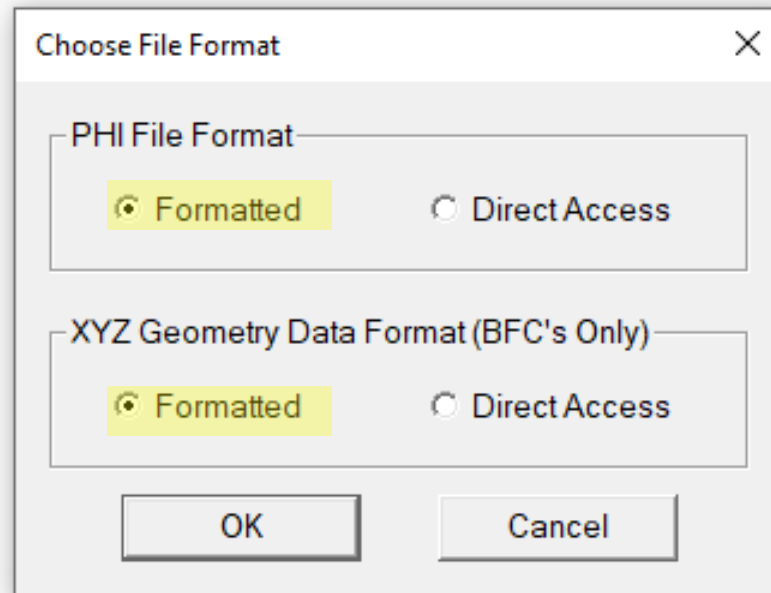
Prof. Gianandrea Vittorio Messa



**POLITECNICO**  
MILANO 1863

# Data formatting

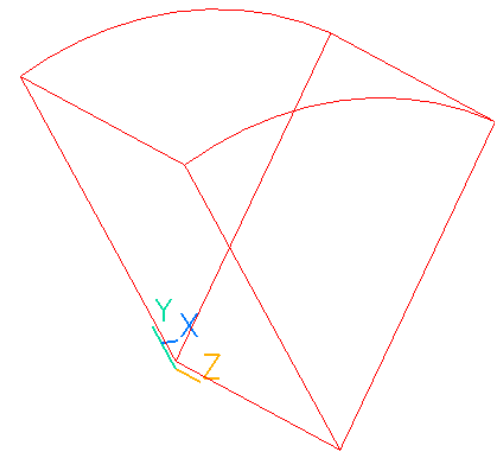
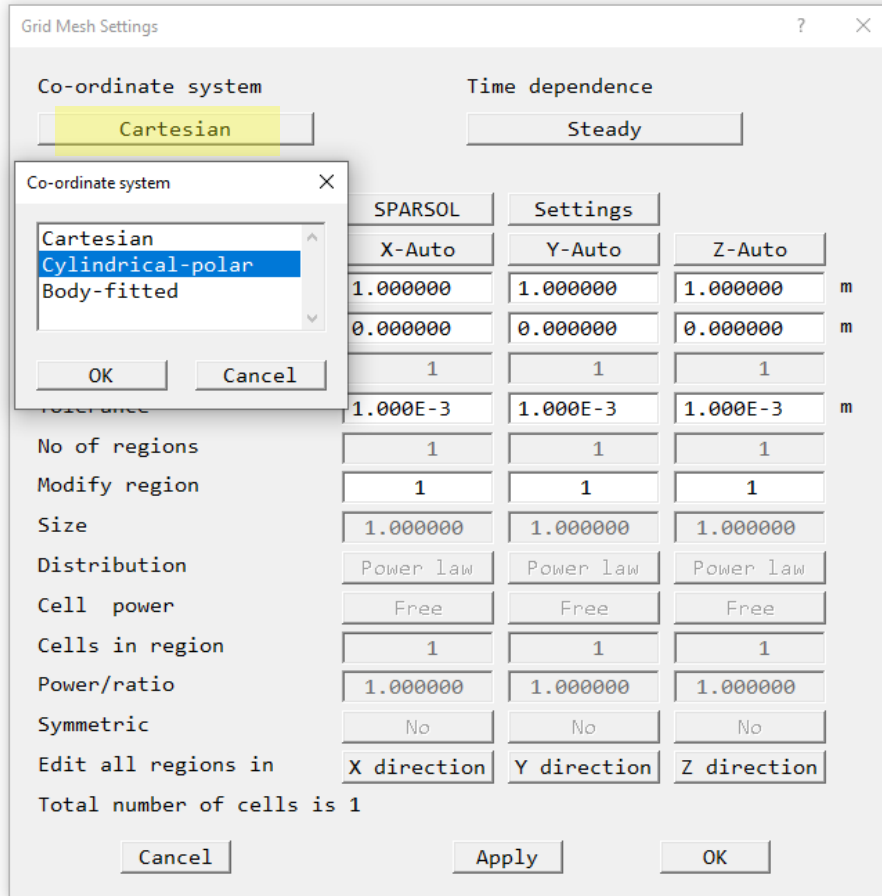
Go to *Options* → *File Format*. Set both queries to *Formatted* for use of post-processing Matlab codes.



# Geometry

## Coordinate system

Go to *Settings* → *Domain Attributes* → *Geometry* → click on «*Cartesian*» and select *Cylindrical-polar*



$$X = \theta, Y = r, Z = z$$

# Geometry

## Domain size

In the same panel, set the dimensions of the numerical domain (X is in rad)

Grid Mesh Settings

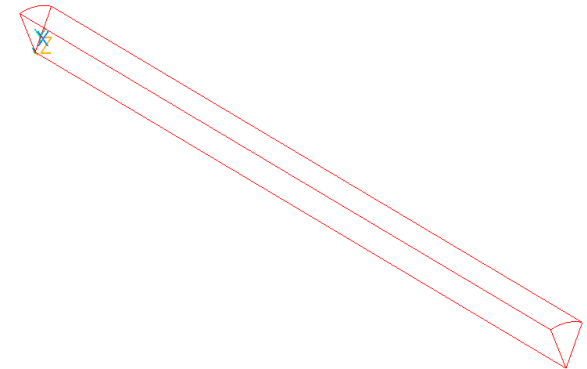
Co-ordinate system: **Cylindrical-polar** Time dependence: **Steady**

Inner radius: 0.000000 m

Cut-cell method: SPARSOL Settings

	X-Auto	Y-Auto	Z-Auto	
Domain size	0.520000	0.075000	???	m
Domain origin	0.000000	0.000000	0.000000	m
Number of cells	1	1	1	
Tolerance	1.000E-3	1.000E-3	1.000E-3	m
No of regions	1	1	1	
Modify region	1	1	1	
Size	0.520000	0.075000	???	
Distribution	Power law	Power law	Power law	
Cell power	Free	Free	Free	
Cells in region	1	1	1	
Power/ratio	1.000000	1.000000	1.000000	
Symmetric	No	No	No	
Edit all regions in	X direction	Y direction	Z direction	
Total number of cells is 1				

Cancel Apply OK

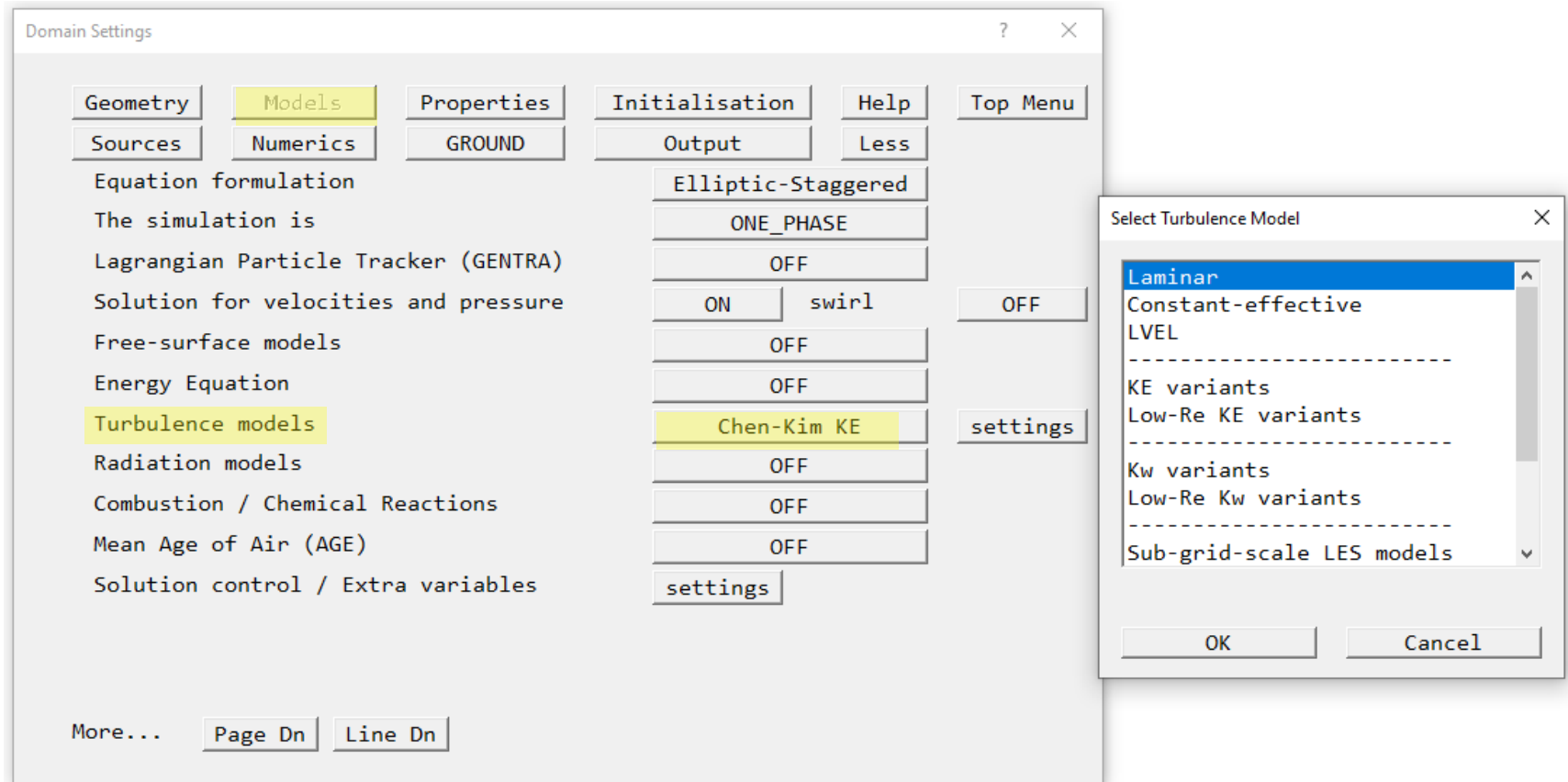


N.B. Axi-symmetry is automatically imposed by PHOENICS

# Models

## Turbulence model

Go to *Settings* → *Models* → *Turbulence models*. Click on the default model and select *Laminar*.



# Fluid properties

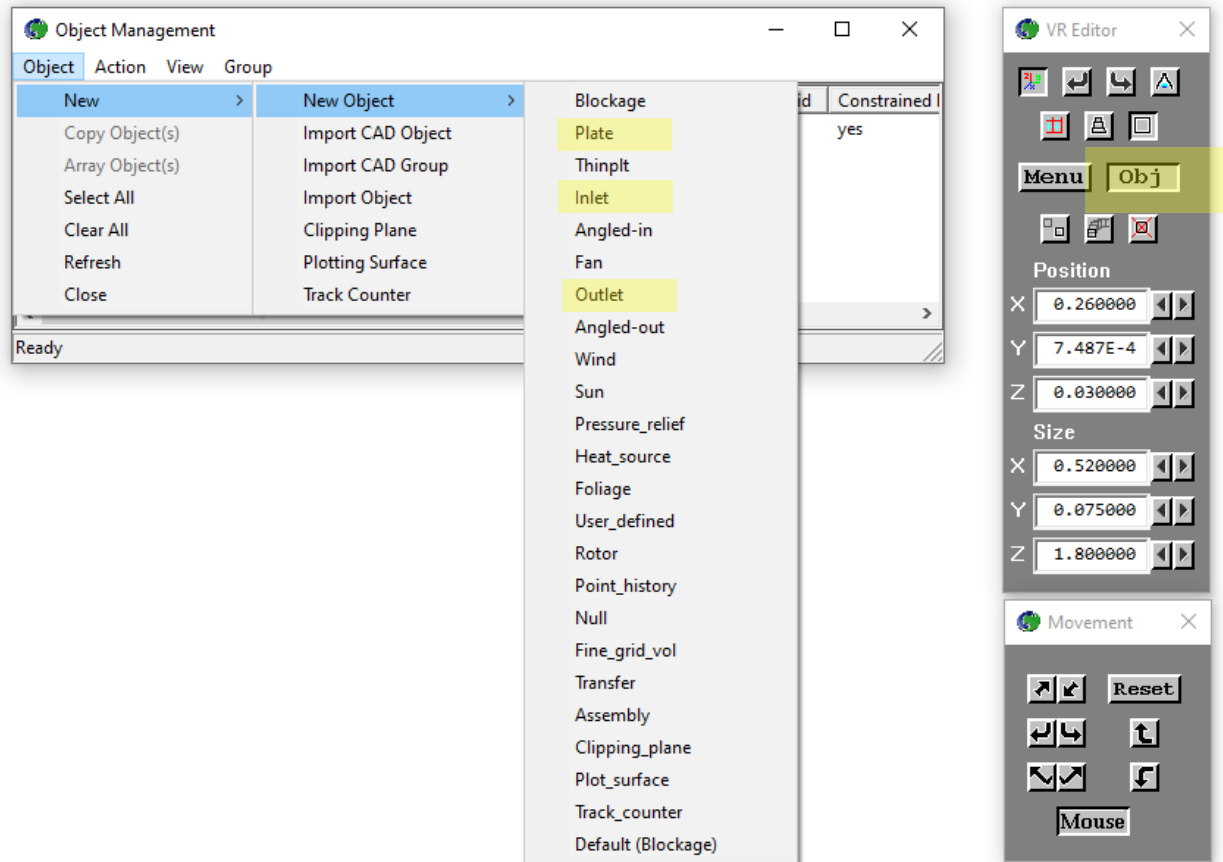
Go to *Settings* → *Properties* → Switch OFF *Use property tables*. Set the density to 910.0 kg/m<sup>3</sup> and the kinematic viscosity to 0.00035 m<sup>2</sup>/s.

The screenshot shows the 'Domain Settings' dialog box with the 'Properties' tab selected. The 'Use property tables' option is set to 'OFF'. The 'Density' is set to 'CONSTANT' with a value of '910.' entered in the 'RHO=' field. The 'Viscosity' is set to 'CONSTANT' with a value of '3.5E-04' entered in the 'ENUL=' field. The 'Temperature' is set to 'NOTSET'. The 'Storage' options for Density, Viscosity, and Temperature are all set to 'OFF'. The 'More...' button is visible at the bottom left, along with 'Page Dn' and 'Line Dn' buttons.

Property	Model	Value
Density	CONSTANT	910.
Viscosity	CONSTANT	3.5E-04
Temperature	NOTSET	

# Boundary conditions

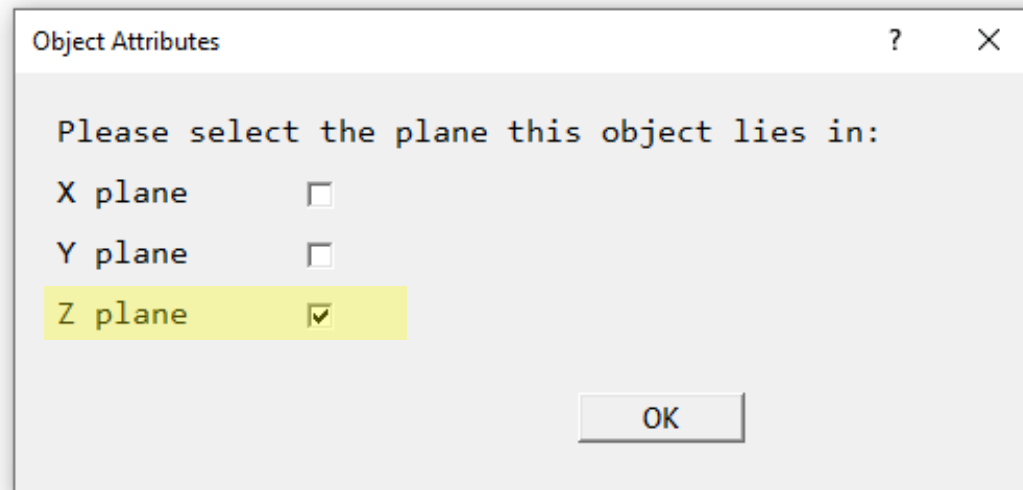
Boundary conditions can be set through *Obj* in VR Editor window. On the *Object Management* window, go to *Object* → *New* → *New Object*; a list will appear.



# Boundary conditions

## Inlet

Select *Inlet* and toggle on the Z plane for its position.

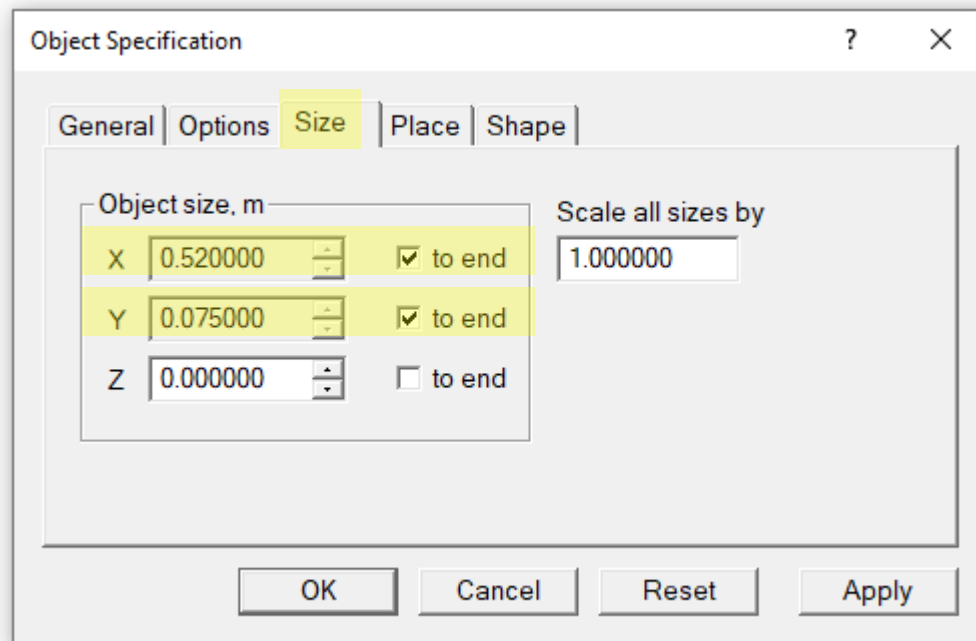




# Boundary conditions

## Inlet

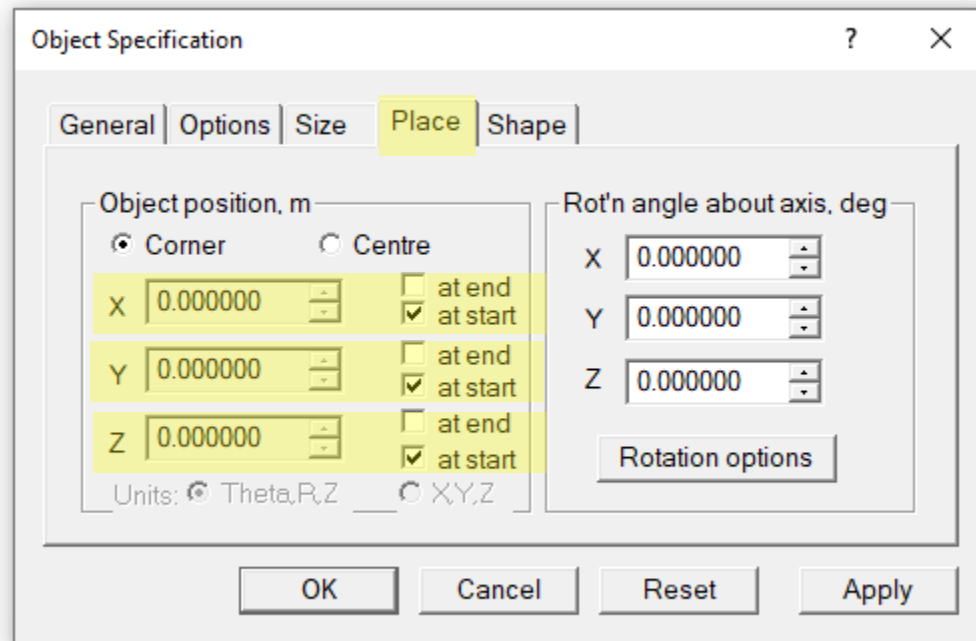
On the *Object Specifications* window, go to *Size*. Set inlet size *to end* for both X and Y directions.



# Boundary conditions

## Inlet

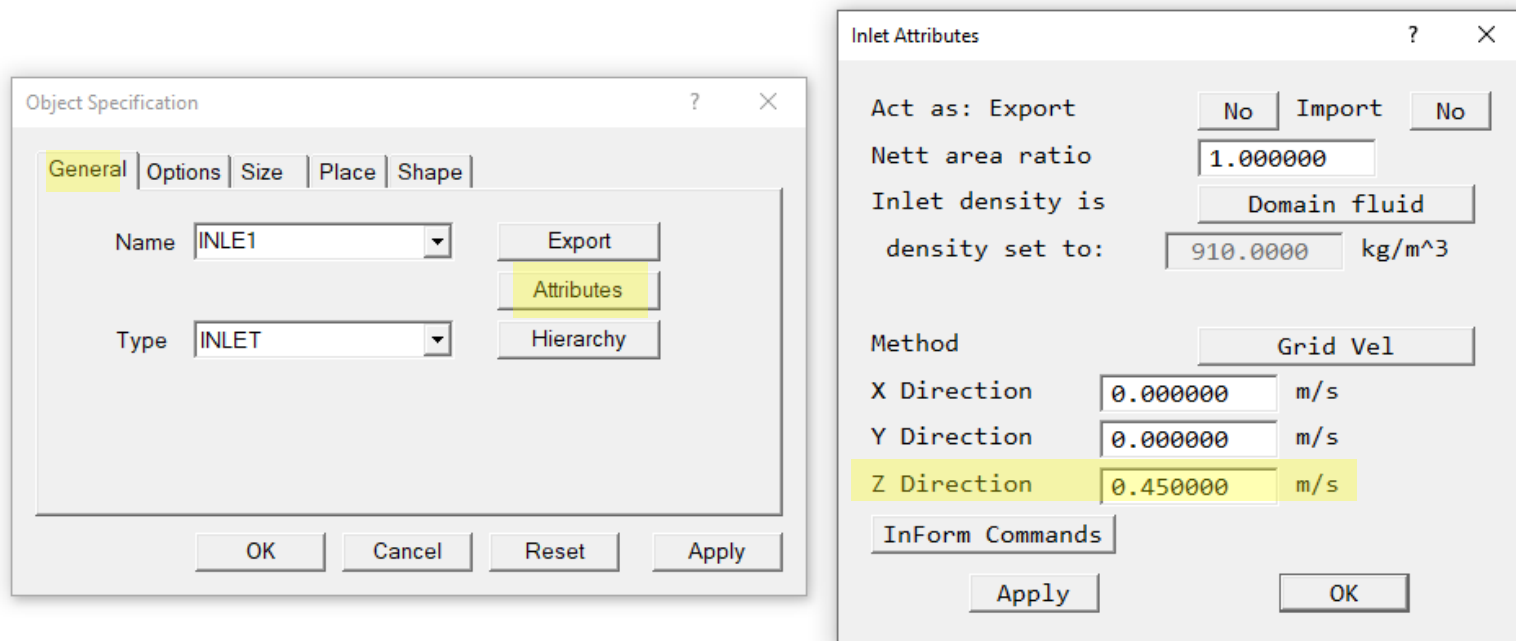
On the same window, go to *place*. Set inlet position *at start* for all dimensions.



# Boundary conditions

## Inlet

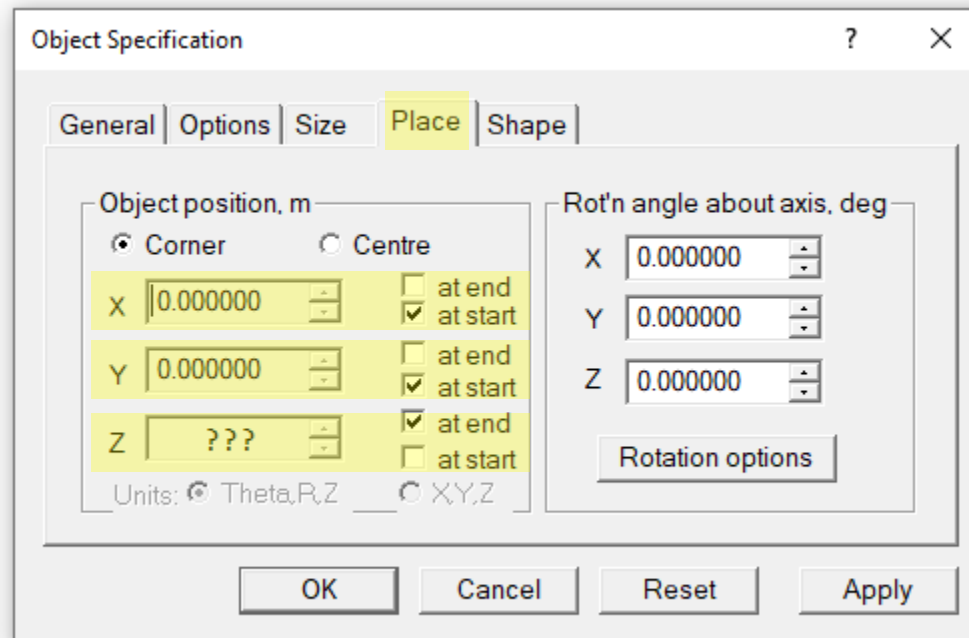
On the same window, go to *General* → *Attributes*. Set the axial velocity to 0.45 m/s.



# Boundary conditions

## Outlet

From the *New Object* window, select *Outlet* and repeat the same steps as for the *Inlet* for plane and size. Set the position *at start* for both *X* and *Y* directions. Set the position *at end* for the *Z*-axis:

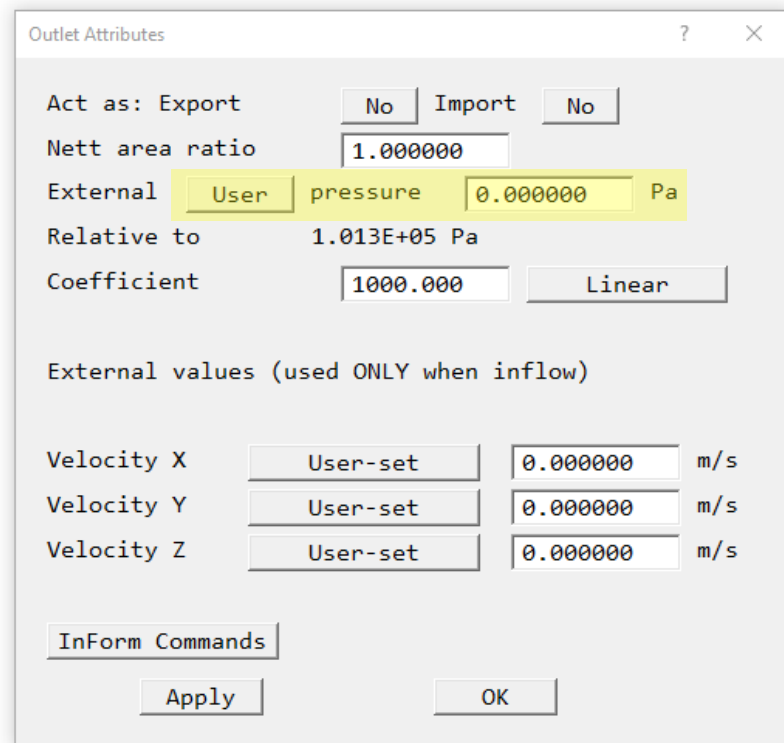
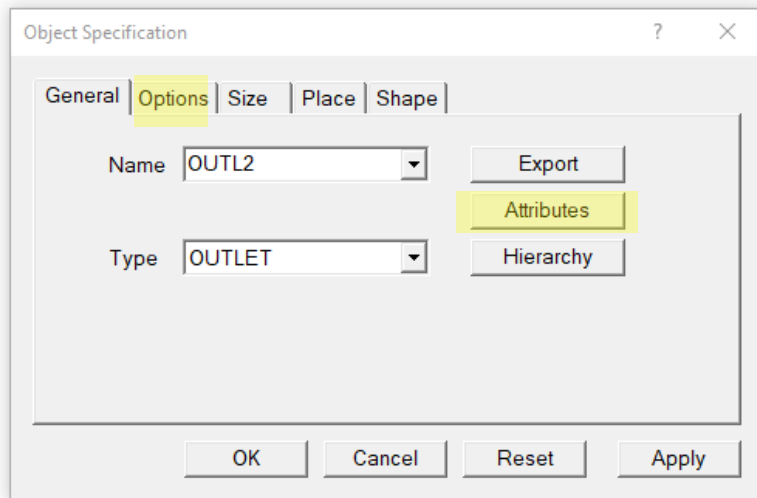


# Boundary conditions

## Outlet

On the *Attributes* window, set the *external pressure* to *user* and impose  $0.000000$  Pa.

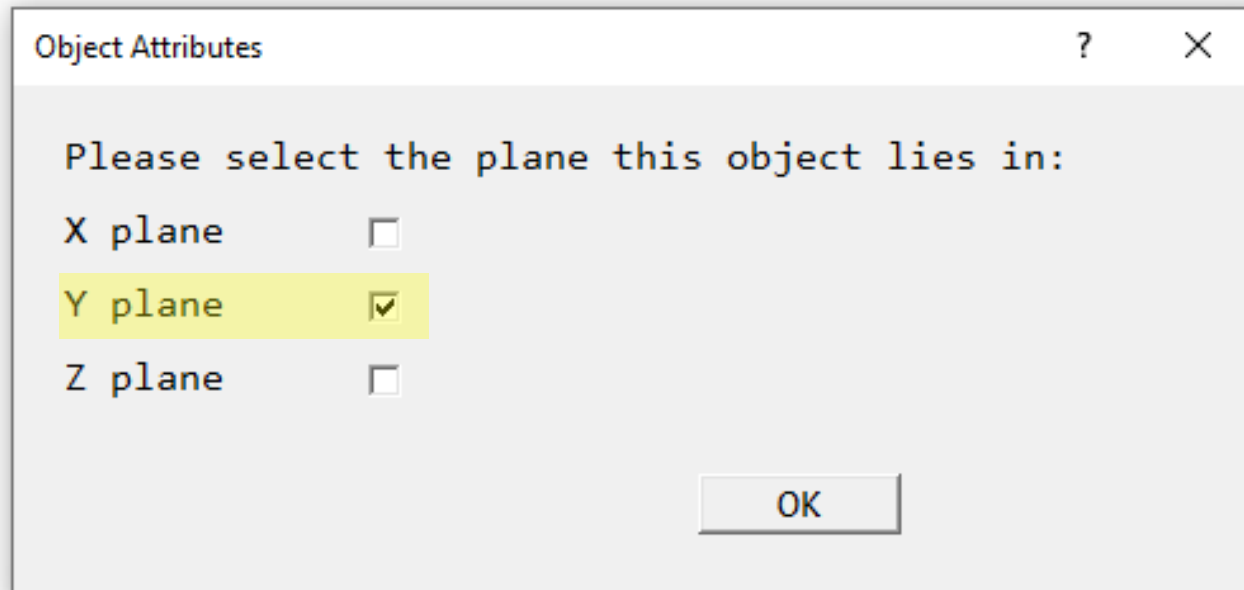
**Suggestion for individual study: how should you change the coefficient to have a null outlet pressure?**



# Boundary conditions

## Wall

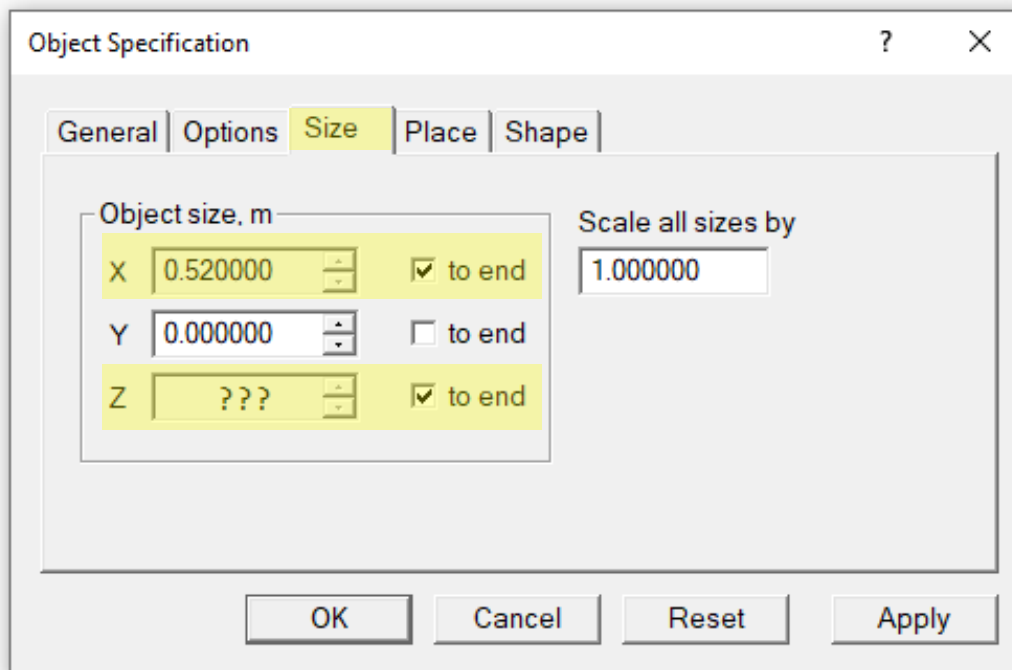
On the *New Object* window, select the *Plate*. The *Plate* object has a zero thickness in the Y plane.



# Boundary conditions

## Wall

On the *Object Specification* of the plate, set its size to the end for the X and Z directions.



# Boundary conditions

## Wall

On the *Object Specification* of the plate, set the *Place* at the end of the Y direction. Set the position *at start* for both X and Z directions.

The screenshot shows the 'Object Specification' dialog box with the 'Place' tab selected. The 'Object position, m' section has 'Corner' selected. The X, Y, and Z coordinates are 0.000000, 0.075000, and 0.000000 respectively. The 'at end' checkbox is checked for Y, and 'at start' is checked for X and Z. The 'Rot'n angle about axis, deg' section has all angles set to 0.000000. The 'Units' section has 'Theta,R,Z' selected. The 'Rotation options' button is visible.

Axis	Position (m)	at end	at start
X	0.000000	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Y	0.075000	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Z	0.000000	<input type="checkbox"/>	<input checked="" type="checkbox"/>

Units: ☒ Theta,R,Z ☐ X,Y,Z

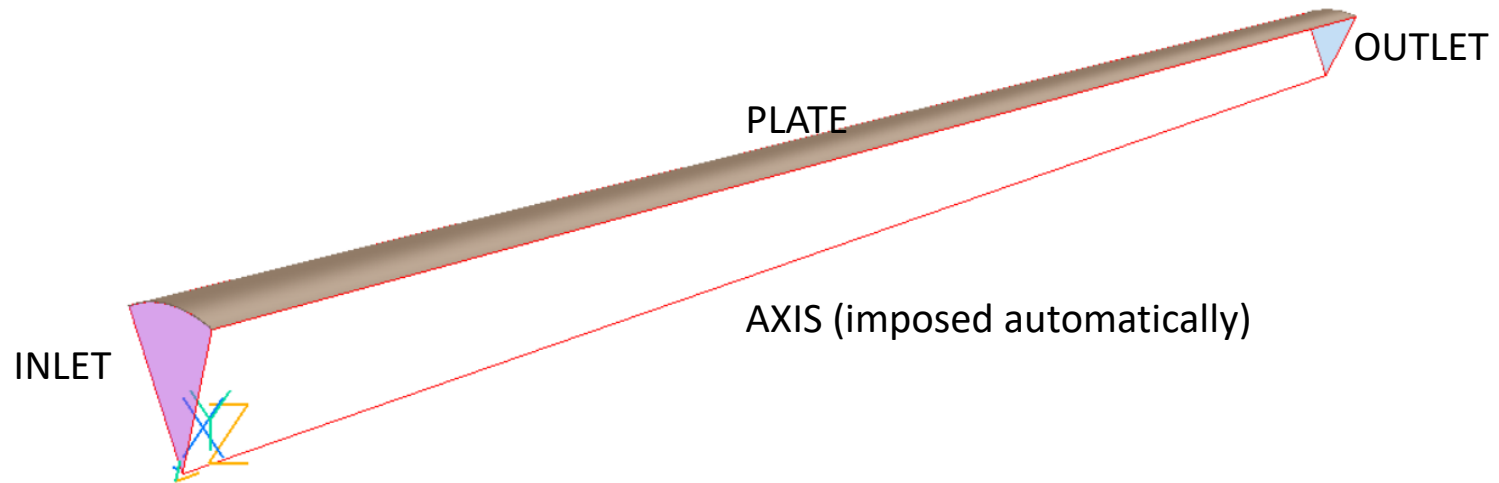
Rotation options



# Boundary conditions

## Wall

The *objects* are visible with different colors.



# Mesh

## Number of cells

In the *Geometry* panel, uncheck *X-Auto*, *Y-Auto* and *Z-Auto* and set it to *-Manual*. Set the number of cells.

Grid Mesh Settings ? X

Co-ordinate system: Cylindrical-polar

Time dependence: Steady

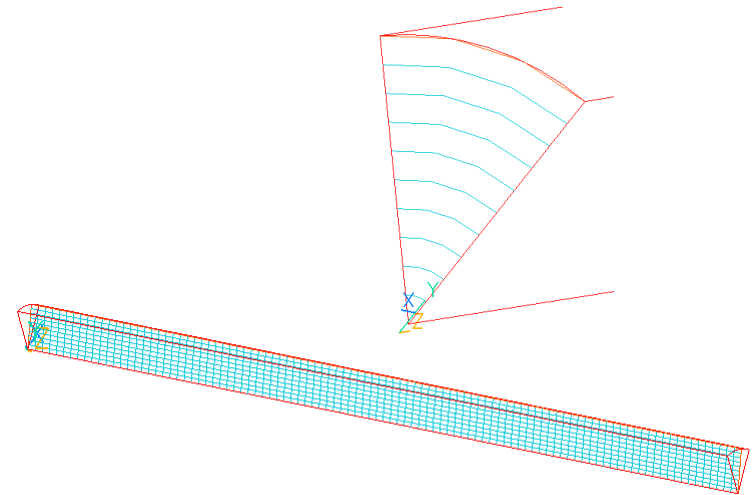
Inner radius: 0.000000 m

Cut-cell method: SPARSOL

	X-Manual	Y-Manual	Z-Manual
Domain size	0.520000	0.075000	???
Domain origin	0.000000	0.000000	0.000000
Number of cells	1	10	100
Tolerance	1.000E-3	1.000E-3	1.000E-3
No of regions	1	1	1
Modify region	1	1	1
Size	0.520000	0.075000	???
Distribution	Power law	Power law	Power law
Cell power	Set	Set	Set
Cells in region	1	10	100
Power/ratio	1.000000	1.000000	1.000000
Symmetric	No	No	No
Edit all regions in	X direction	Y direction	Z direction

Total number of cells is 1000

Cancel Apply OK



N.B. Axi-symmetry is automatically imposed by PHOENICS. No need to discretize in X.

# Mesh

## Power law

Cells can be distributed non-homogeneously along a direction through a *power law*.

Grid Mesh Settings

Co-ordinate system: Cylindrical-polar

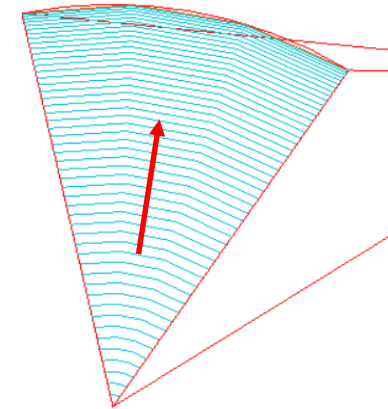
Time dependence: Steady

Inner radius: 0.000000 m

Cut-cell method: SPARSOL

	X-Manual	Y-Manual	Z-Manual	
Domain size	0.520000	0.075000	???	m
Domain origin	0.000000	0.000000	0.000000	m
Number of cells	1	10	100	
Tolerance	1.000E-3	1.000E-3	1.000E-3	m
No of regions	1	1	1	
Modify region	1	1	1	
Size	0.520000	0.075000	???	
Distribution	Power law	Power law	Power law	
Cell power	Set	Set	Set	
Cells in region	1	50	100	
Power/ratio	1.000000	-1.500000	1.000000	
Symmetric	No	No	No	
Edit all regions in	X direction	Y direction	Z direction	
Total number of cells is				

Buttons: Cancel, Apply, OK



Decreasing height along Y

# Numerics

## Number of iterations

From *settings* → *domain attributes* → *numerics*, set the maximum number of iterations and a global convergence criterion.

Domain Settings ? ×

Geometry Models Properties Initialisation Help Top menu  
Sources Numerics GROUND Output Less

Total number of iterations

Minimum number of iterations

Maximum runtime  Limit  s

Global convergence criterion  %

Relaxation control Iteration control

Limits on Variables Differencing Schemes

Advanced settings

# Note

## Calculation of the shear stress

The shear stress profile in the fully developed region can be calculated through the following formula

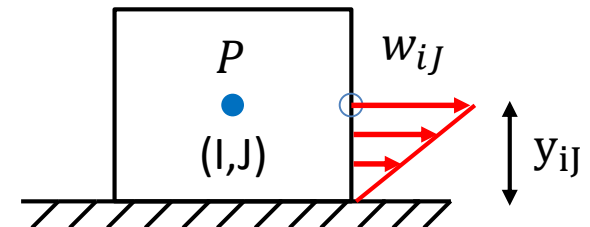
$$\tau_{rz}(r, \theta, z) = -2\mu \frac{1}{2} \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial r} \right) = -\mu \frac{dw}{dr}$$

The numerical calculation of the derivative  $dw/dr$  can be done either

- 1) by referring to the PHOENICS variable DWDY (note that, in PHOENICS, the variable DWDY is stored in the cell centres)
- 2) by approximating the derivative starting from the W1-distribution (pay attention to how the derivative is approximated and where the derivative is stored according to the selected scheme!)

Note that, in laminar flow, the W1 is assumed linear between the wall and the first grid node, therefore

$$\tau_w^{CFD, lam} \approx \mu \frac{W_{iJ}}{y_{iJ}}$$



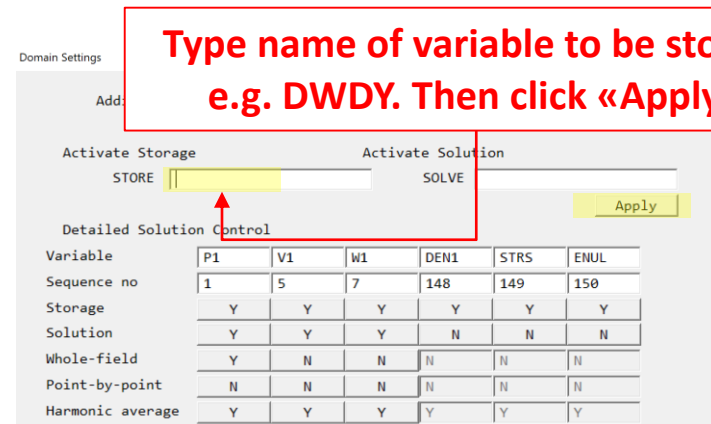
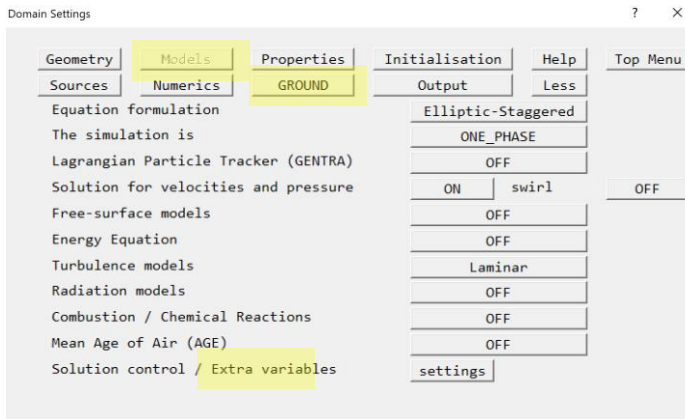
# How to store additional variables

Users can store all variables in the following list

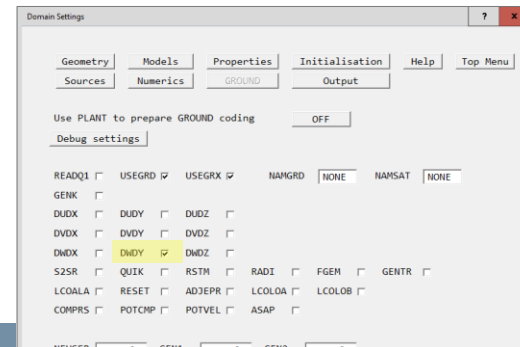
[https://www.cham.co.uk/phoenics/d\\_polis/d\\_enc/reserv.htm](https://www.cham.co.uk/phoenics/d_polis/d_enc/reserv.htm)

The list includes, for instance, the variables DWDY and DWDZ.

In order to store these variables in the .phi solution file, go to *Models > Solution control / Extra Variables*



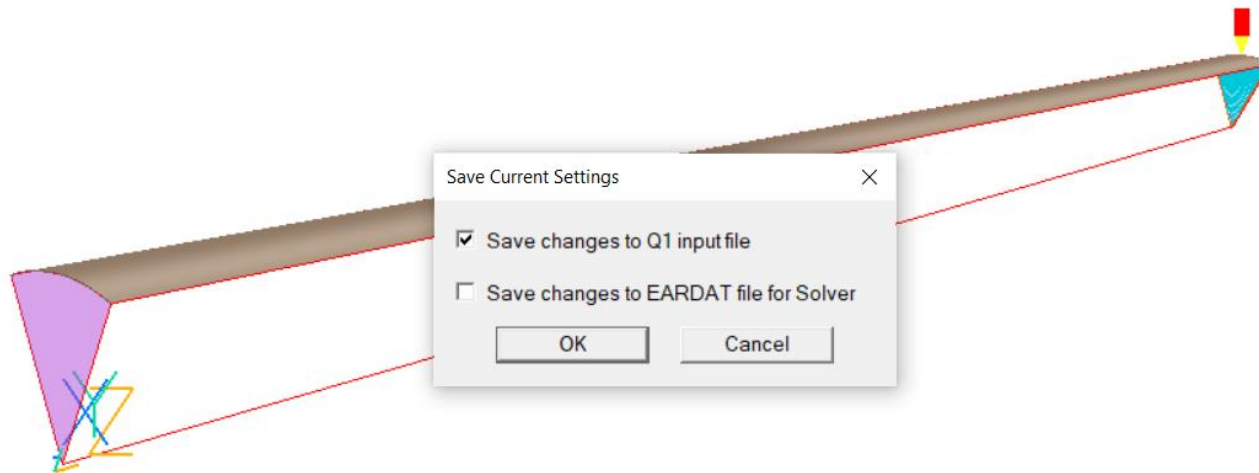
For derivatives, go to *GROUND* and flag the quantity



# Save

## Store q1 file

Go to *File* → *Save As a Case* and toggle on the *Save changes to Q1 input file*. Then select a proper folder for storing the «instruction» *q1* file.



# Run

## Convergence monitoring: EARTH solver

Go to *Run* → *Solver*. Click *Ok* on the pop-up windows. The EARTH solver will open.



Values of the main output variables (P1, V1, W1) at probe position.

Whole field residual for the main control variables (P1, V1, W1)

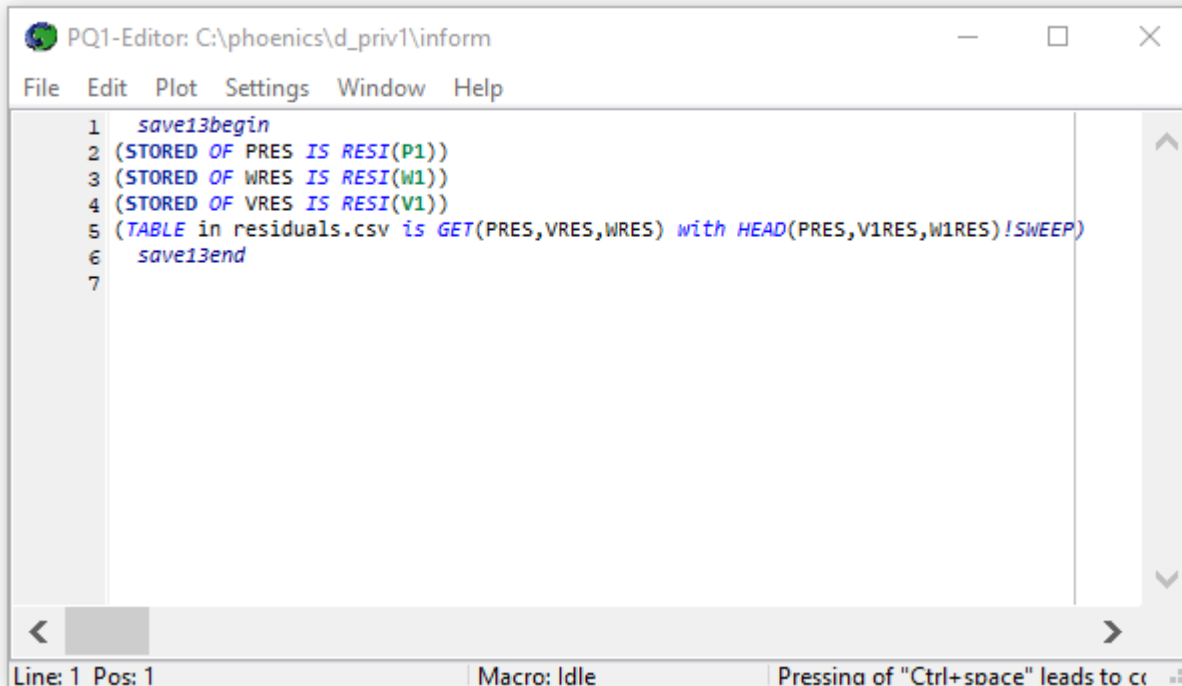
Note: this is the Classic Solver GUI. Go to *Options* → *Solver Monitor Options* → *Monitor GUI Style = Classic*



# Run (optional)

## Convergence monitoring: residuals.csv

It is possible to export residuals to a .csv format through the InForm language. Go to *Settings* → *Domain Attributes* → *INFORM* → In *All Save blocks* select *save13* and press *Edit Save block*. Then insert the following code and *Save*.

The image shows a screenshot of the PQ1-Editor window. The title bar reads "PQ1-Editor: C:\phoenics\d\_priv1\inform". The menu bar includes "File", "Edit", "Plot", "Settings", "Window", and "Help". The main text area contains the following InForm code:

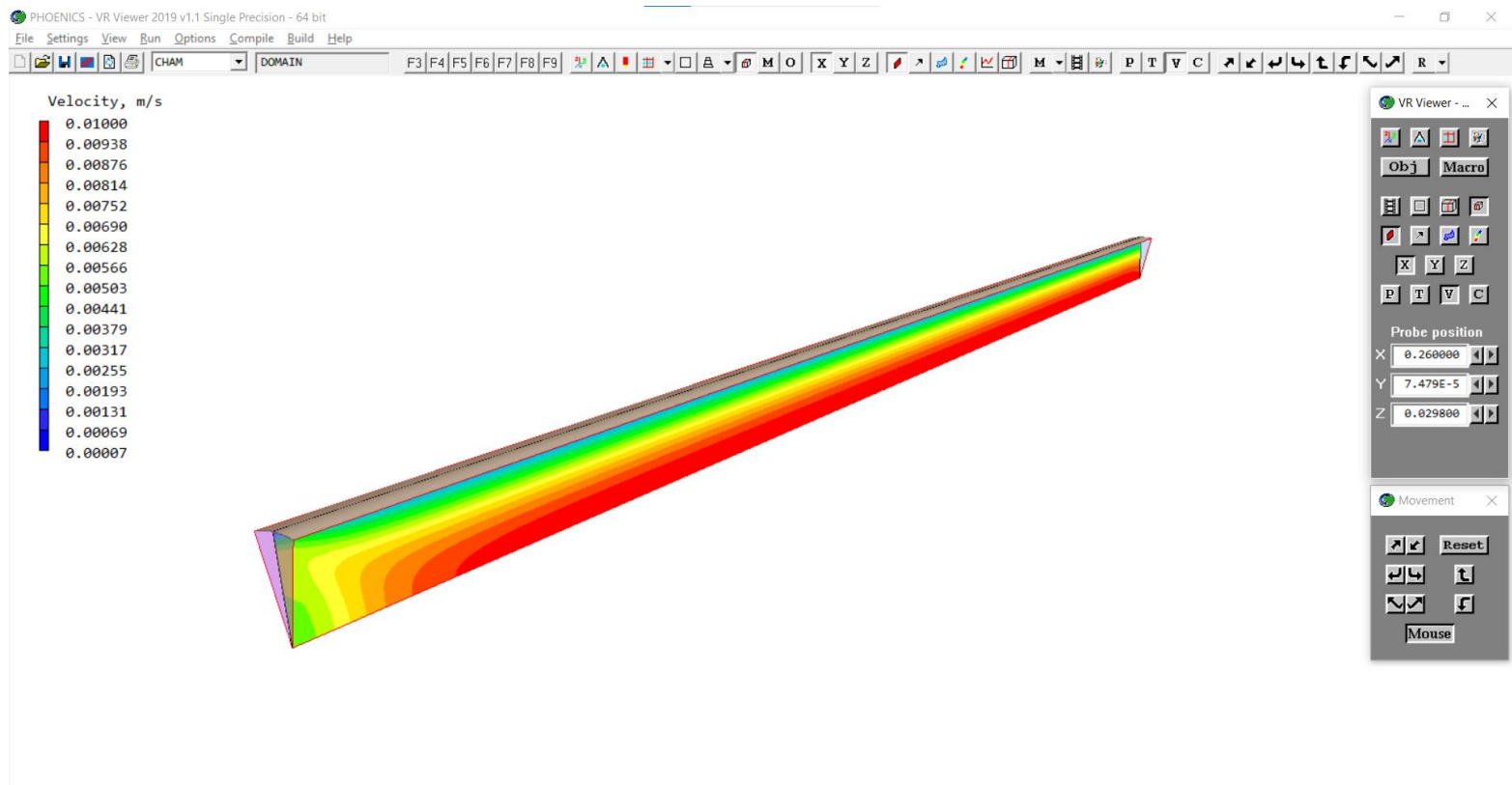
```
1  save13begin
2  (STORED OF PRES IS RESI(P1))
3  (STORED OF WRES IS RESI(W1))
4  (STORED OF VRES IS RESI(V1))
5  (TABLE in residuals.csv is GET(PRES,VRES,WRES) with HEAD(PRES,V1RES,W1RES)!SWEEP)
6  save13end
7
```

The status bar at the bottom indicates "Line: 1 Pos: 1", "Macro: Idle", and "Pressing of 'Ctrl+space' leads to c".

A table named «residuals.csv» will be created in the *dpriv\_1* folder. It will store the residuals for pressure and velocity at each sweep (or iteration).

# Post-processing VR-Viewer

Once the calculation has finished, go to *Run* → *Post Processor* → *GUI Post Processor (VR Viewer)* to preview the results. Select the result *phi* file to visualize the result.



# Post-processing

## Matlab import

In order to answer the questions provided in the *Requests*, it is recommended to use Matlab.

A script for importing formatted files from PHOENICS is provided (*XYZ\_reduced\_19\_20.mat*). The following variables are imported:

- $NX, NY, NZ$  – are the number of cells in X, Y and Z directions
- $P1, V1, W1, \dots$  – are the matrices of the computed variables
- $X_C, Y_C, Z_C$  – are the coordinates vectors of the cells' centers.
- $X_E, Y_N, Z_H$  – are coordinates vectors of the cells' centers.

